Master Designer

Version 8.7

What's New for MD 8.7

Contents

December 1997

Contents

Contents

About This Manual

This manual is for those of you who are upgrading to Master Designer 8.7. It explains all the new features and changes that have been made since Master Designer 8.6. By reading this manual, you'll be able to see at a glance what's new and different in Master Designer 8.7.

This manual consists of four chapters and five appendixes. A conventions page follows this section.

Chapter 1	"What's New" describes what's new throughout Master Designer 8.7, such as automatic assignment of reference designators, Level push/pop into padstacks, and snap to pin in Symbol and Part Editors.
Chapter 2	"What's New in the Utilities" describes what's new in the Master Designer 8.7 utilities, including the reference designator use report, alpha numeric sorting of wire list, and the reporting of unassigned pins shorted to inner layer planes
Appendix A	"System Limits" describes name length limits; maximums for database items such as pins, components, and nets; and ranges for other items.
Appendix B	"Filename Extensions" lists the P-CAD filename extensions and the tools that produce the files.
Appendix C	"Reserved Words" lists DOS reserved device names and P-CAD attribute keywords.
Appendix D	"Button Menu Tree" contains a graphical menu tree for the Button Menus.
Appendix E	"Command Cross Reference" contains a command cross-reference, a list of default hot-key functions, and a chart of the keys available for hot-key assignment.

Conventions

This manual uses the following conventions:

→ or !	Connects commands. Commands following the arrow appear on submenus. For example *File*!*Load* or *File*Õ*Load*
*	One or more characters can occupy the asterisk's position. Also known as a wildcard.
*	In text, introduces a procedure that explains how to do a task.
[ ]–[ ]	Press the keys simultaneously. For example [Shift]–[F1]
[↵]	The return key. Press this key after typing data in a data entry box or on a message line or to accept a default. You can click left in many P-CAD tools instead of pressing [↵].
Enter or ↵	Indicates you need to press [↵] after typing data.
[Space]	The space bar. You can use this key to digitize a point within the drawing area.
italics	Indicates variable characters.
boldface	Indicates characters you enter from the keyboard; for example Enter **sheet7** in the data entry box.

boldface italics	Indicates variable characters you enter from the keyboard; for example
	Enter ***filename*** in the data entry box.
Click left	Press and release the left button (button 1) on the mouse.
Click middle	Press and release the middle button (button 2) on the mouse.
Click right	Press and release the right button (button 3) on the mouse. If you are using a two-button mouse, press Ctrl and button 2 simultaneously to clickright.
Cycle	Click left repeatedly on a cycle box until the item you want is selected. A cycle box is indicated by ◇.
Select	Move the cursor to an item or point and press ↵, Space, or click left.

What's New 1

P-CAD Master Designer 8.7 (MD 8.7) gives you the power and flexibility to create electronic circuit designs and PCB (printed circuit board) layouts from start to finish. With its many enhancements, however, MD 8.7 lets you complete your designs even faster than before.

Graphic Editors

Zoom controls using the unshifted "+" and "-" keys

To improve the capability to view areas of the design file at various zoom levels, two additional hot keys have been added to the zoom controls:

- unshifted "+" (plus) key - used to zoom in on a area of the design.
- unshifted "-" (minus) key - used to zoom out.

The zoom function remains unchanged from previous versions of the product, the addition of the unshifted "+" and "-" keys are provided to allow greater flexibility in how the zoom controls are accessed.

Snap to Object in Measure

Master Designer 8.7 now allows the capability of snapping to off-grid design objects when using the *Measure* command. The snap to object feature is available in all Editors (Symbol, Parts, Schematic, and PCB) and can be enable or disabled using the new */rsnp* keyboard command.

To use the snap feature for Measure

1. From any Editor, select *Display→Measure.* The system prompts

   Select point to measure from

2. Move the cursor near the location of an object from which you wish to begin measuring.

3. At this point the System will display, on the status line, the RX, RY, and Actual measurements from the beginning object to the current cursor location.

4. Enter Shift - s

Note: In order to snap to objects, insure that Caps Lock is not enabled.

2. The cursor will snap to the nearest object and this will be the start point for *Measure*.

3. Move the cursor near the location of an object from which you wish to end measuring.

4. Enter Shift - **s**.

5. The cursor will snap to the nearest object and this will be the end point for *Measure*.

6. The status line will now display the RX, RY, and Actual Distance between the two points.

The addition of the snap feature allows snapping to the following objects:

Drawn Line
Drawn Arc
Drawn Circle and Filled Circle
Drawn Rectangle and Filled Rectangle
Drawn Polygon
Entered Polygon
Polygonal Void
Circular Void
Wire segment
Curve wire
Pin center location if no padstack attached
Via center location if no padstack attached
All flash shapes except Thermal, Special, and Target
Center points of Thermal, Special, and Target flash shapes
Bounding box of drawn text
Stand alone flash center location (Part Editor Only)

Note: Snap to object will not snap to free standing flashes in the PCB Editor.

If there are more than one object within snap tolerance, the program will snap to the object closer to the cursor. If two or more objects within snap tolerance are equally distant to the start point, the program will snap to the object with higher layer priority. The priorities of layers are:

1. Active
2. Able
3. On

If two or more objects have the same distance and the same layer priority, the program will snap to the one first found in the database.

When snapping to an object, the program can actually snap to the center, the edge, or the vertex of a line or an arc depending on user option. The */rsnp* will let you set this option. When using the */rsnp* you will be presented with a new dialog box (see the figure below) which will present you will three options.

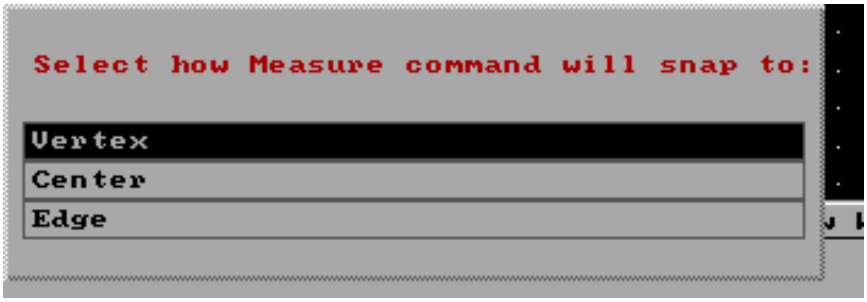


Figure 1.1

The three available options are described below:

- **Vertex** - The measure command will snap the cursor to the nearest vertex. This is the default setting and will always be selected until changed by the user.

- **Center** - The measure command will snap the cursor to the center of a line or arc.

- **Edge** - The measure command will snap the cursor to the edge of a line, arc, or circle, etc..

Symbol Editor and Part Editor

Snap to Pin for entering pin sequence, naming pins, query pin, and editing pin types

The Enter Pin Sequence, Name Pin, Query Pin, and Edit Pin Type commands have been improved to allow the automatic snapping to off-grid pins. This provides improved productivity by allowing a greater latitude in pin selection, especially for fine pitch devices or those devices having inconsistent pad spacing.

The Pin Sequence and Name Pin operations will now provide the capability to snap the cursor to the center of the closest pin, rather then the current requirement of placing the cursor at the exact pin center. The current snap tolerance will be used to determine what pin is being selected. If the point selected is outside of the pin's current snap tolerance, "No Pin Found" will be displayed on the status line.

If the point selected is within the snap tolerance of two pins, the cursor will snap to the closest pin center. If the point selected is within the snap tolerance of two pins and an equal distance between both pin centers, the Pin Sequence command will attempt to determine the direction of sequencing and select the pin that should be sequenced next. If this is not possible, than the next pin in the database will be selected.

PCB Editor

Level Push and Level Pop on Padstacks

MD 8.7 now offers the capability to allow Level Push and Level Pop commands to operate on a pin or via Padstack, similar to the Symbol and Part Level Push and Level Pop commands.

A new subcommand "Push Padstack" has been added to the File menu. When this command is invoked you will be prompted to select a pin or via. A selection of a valid pin or via, with an attached padstack, results in a level push into the padstack. An invalid selection causes the PCB editor to display an "No pin or via found" error message on the status line and you are prompted to make another selection.

When pushing into a Padstack, the PCB Editor will change from the PCB Editor to the Part Editor and load the padstack file (.PS) for the pin or via selected. The user can then return to PCB Editor by using the Level Pop command. If the loaded padstack was modified prior to selecting Level Pop, the user will be prompted to save the edited padstack and, if saved, replace all occurrences in the existing design file.

The padstack can only be saved under it's original name. If you need to modify a padstack and save it under a different name, then the Part Editor should be used, rather then *File→ Push Padstack.*

Note: If you are using a System Directory configuration, the *Push Padstack* command will search for padstack files in the current System Directory, unless the Special Symbol File uses pathing to indicate the location of the padstacks. If the selected padstack file can't be located by either of these methods, a "padstack.ps not found" error message will be displayed on the status line. To locate the proper padstack file you will need to identify the System Directory used to initially attach the padstack to the database.

Maintain link with padstack files when "Cancel" is selected from Attach Padstacks

Master Designer 8.7 now allows the link to existing padstacks to be maintained when selecting the Cancel button during the Environment Attach Padstacks command. The cancel button will now allow you to cancel the Environment→Attach Padstacks command and select a new command, without removing the existing link to attached padstacks.

Note: To remove the link to the existing padstacks in the current design, use the backspace key to highlight the current Special Symbol File filename, and then enter the return key. This will erase the current Special Symbol File filename and unlink the current padstack files.

Schematic Editor

Automatic Assignment of Reference Designators

Master Designer 8.7 can now automatically assign reference designator and section numbers to symbols as you enter them in the Schematic Editor. The system also remembers the last reference designator and section number entered for each symbol. This feature is useful because it saves you from having to manually specify the reference designator and section numbers after entering the symbol. You also don't have to worry about remembering the last reference designator and section numbers entered. Master Designer assigns and increments them automatically.

The automatic assignment of reference designators is accomplished based on the PREFIX attribute assigned to the symbol. You can assign the PREFIX attribute to symbols individually or a utility introduced in MD 8.7 can be used to assign the PREFIX attribute to all symbols in a directory, or all symbols in a library (.SLB). For a more detailed explanation on how to use the utility for assigning the PREFIX attribute see Assign Reference Designator Prefix under Prevue in What's new in Utilities.

To assign the PREFIX attribute to an individual symbol

1. From the Symbol Editor, select *File→Load*. The system prompts

   Enter file name:

2. Enter the name of the desired component, or select *PICKLIST* to choose the component from the picklist. Remember that you can change directories if the desired component is in a different directory.

3. Click *ENTER*.

4. Select *Enter→Attribute*. The system prompts

   Select location (Text attributes OK?)

5. Select the location for the new Attribute. The system prompts

   Type in attribute spec.

6. Enter **prefix = * ;** where * indicates the reference designator prefix that you would like to assign to the symbol.

7. Select *File→Save* the symbol.

Note: For symbols in an existing schematic you can use the *File→Level Push* and *File→Level Pop* commands to add a PREFIX attribute to individual symbols.

To automatically assign reference designators

1. From the Schematic Editor, Select *Enter→Component*. The system prompts

   Enter component name:

2. Enter the name of the desired component, or select *PICKLIST* to choose the component from the picklist. Remember that you can change directories if the desired component is in a different directory.

3. Click *ENTER*.

   After you enter the symbol name in the response area or choose a symbol from the picklist, the system displays the component outline. Master Designer also displays a *Refd* checkbox in the status area and prompts

   Select location to place *filename*. (Orientation OK?)

4. Change the status area options as needed.

5. Click the *Refd* checkbox. A button appears next to the *Refd* checkbox, which displays the next available Reference Designator and section number. This is the Reference Designator button, which you can use to assign a reference designator and section number to a component that is different then the next sequential assignment.

Note: If you enter a reference designator and section number that is different than what is indicated as being the next available, all subsequent assignments will begin from the reference designator and section number that you enter. Skipped reference designators and section numbers can be assigned by entering the reference designator and section number for those that have been skipped. The next available reference designator and section numbers will increment to the next available skipped gate until all skipped gates

are assigned and then incrementing will continue after the highest reference designator and gate section assigned.

7. Select the location of the component. Master Designer places the component and displays its reference designator and pin number information.

8. Repeat step 7 for each component you want to place. As you place each component, the system increments the Reference Designator button, indicating the reference designator and section of the component you're about to place.

9. Click *CANCEL* or select another command when done.

Prevue

Assign Reference Designator Prefix Attribute

A new utility has been added to Master Designer 8.7 that provides the capability to add a new PREFIX attribute to **All** symbols in a user specified directory or **All** symbols in a user specified library file (.SLB). The Assign Reference Designator Prefix utility is available from the *Library Manager* area of the *Prevue* and will automatically locate all of the symbols, determine the correct symbol/part mapping and use the part's SCAT value to determine the Ref Des prefix to be assigned to the symbols. The directory and/or symbol libraries (.SLB) to be processed are determined by the symbol *search path* and the libraries specified in *Set Libs. & Search Paths* settings in *Prevue*. For an example of the location of the Assign Ref Des Prefix utility see the figure below:

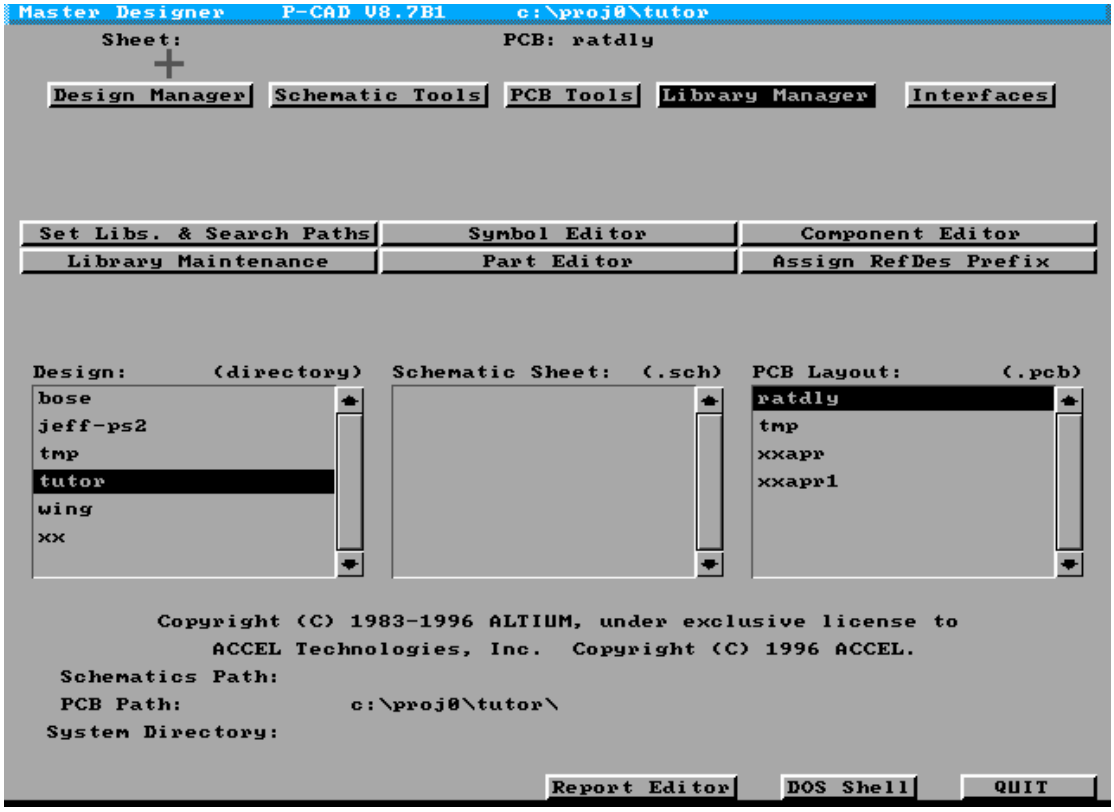


Figure 1.2

Note: The Assign Ref Des Prefix utility will use one of three ways to accomplish finding part-symbol match. It first searches for a part attribute on the symbol. If it can't find part attribute, it will read the cross-reference file (.FIL) to see if the symbol name is included. If a match still can not be made after trying first two approaches, the program will use "same name" method to determine the part file to be used. If a prefix definition exists in the cross reference file, the Assign Reference Designator Prefix utility will use that information, rather then the SCAT value to define "PREFIX" attribute. The PREFIX attribute will be added on the "ATTR2" layer. The layer visibility of "ATTR2" will not be changed because of adding "PREFIX" attribute.

The Assign Ref Des Prefix dialog allows you to confirm search paths, part libraries, and symbol libraries. If any of the settings need to be changed, this can be accomplished by using *Library Managers*, *Set Libs & Search Paths.* For an example of the Assign Ref Des Prefix dialog, see the figure below:

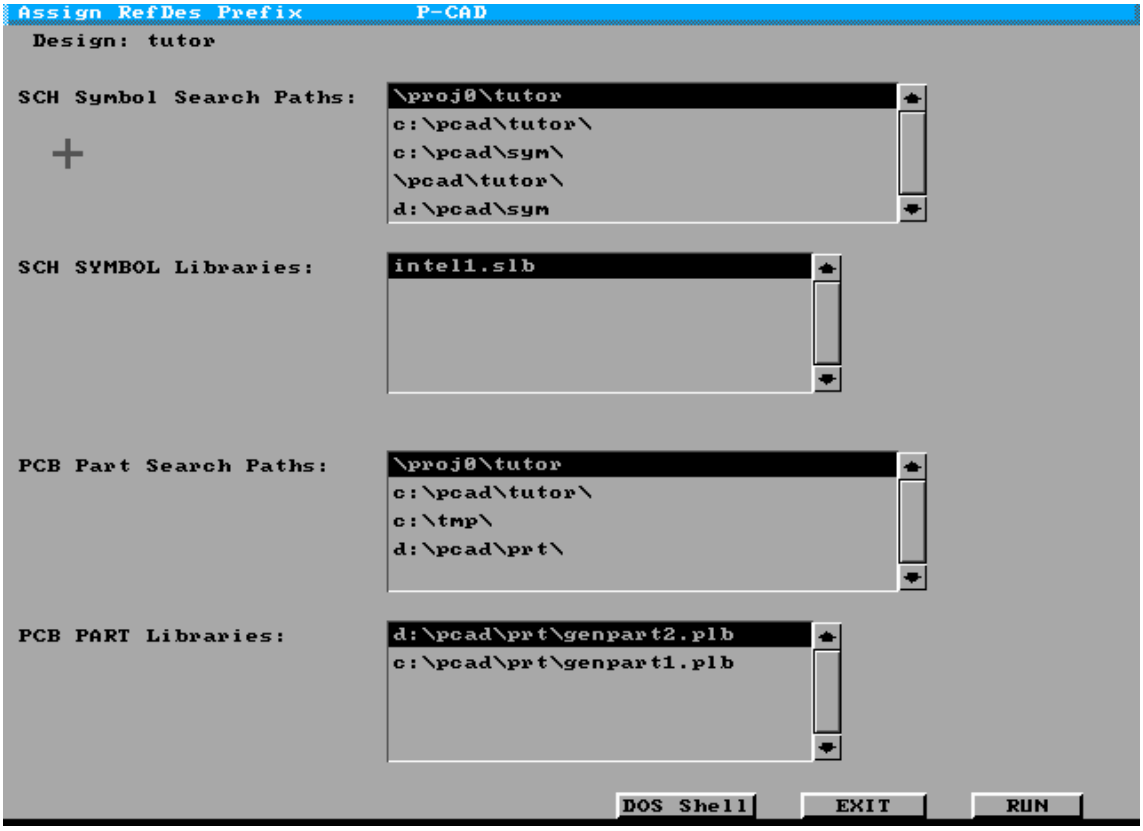


Figure 1.3

Note: All warning or error messages will be listed in the PCAD.LOG file.

What's New in the Utilities 2

This chapter describes the enhancements to the following MD 8.7 utilities

- Design Rules Checking
- Report Generator
- Autorouter

General

Recognition of "." (period) as a valid character in PWGD attribute values

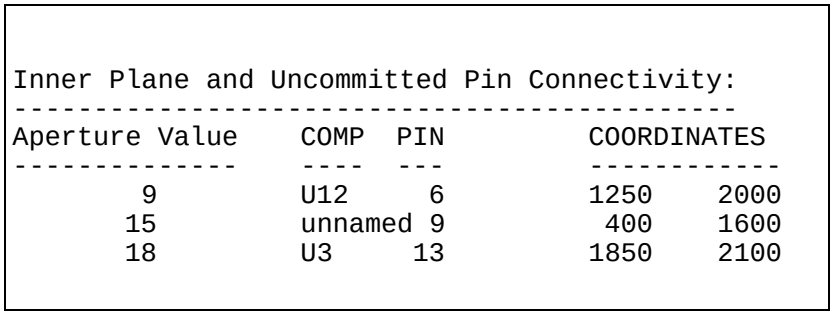
All utilities in Master Designer 8.7 now recognize the use of a "." (period) in PWGD attribute values or the PWGD information defined in the Pin/Net section of the Cross-Reference file, as a valid character. In the case where PWGD information is defined in both the PWGD Attribute of a schematic symbol and in the Cross-Reference PWGD information for the same symbol, the schematic symbol's PWGD Attribute definition will supersede that of the Cross-Reference file.

Design Rules Checking

Reporting of uncommited pins shorted to inner planes

Design Rules Checking has been enhanced, in MD 8.7, to report uncommitted pins that are shorted to internal planes. Currently DRC can only report a pin shorted to an internal plane if that pin is part of a signal net. Design Rules Checking will take uncommitted pins into account when checking inner plane connectivity.

If an uncommitted pin is found shorted to an inner plane, the DRC Error report will record this information and let the user easily find which pin is causing the short by adding a new section to the DRC report that lists the reference designator, pin number and pin location of the uncommitted pin. The figure below provides an example of the new INNER PLANE AND UNCOMMITTED PIN CONNECTIVITY section included in the Design Rules Error report.



Inner Plane and Uncommitted Pin Connectivity:

Aperture Value	COMP	PIN	COORDINATES	
9	U12	6	1250	2000
15	unnamed	9	400	1600
18	U3	13	1850	2100

Figure 2.1. Inner Plane and Uncommitted Pin Connectivity

Note: For Design Rules Checking of uncommitted pins to occur, the appropriate inner plane Flash layers must be included in the DRC Check Passes.

Report Generator

Alpha-numeric Sorting of Wire List

The *Report Generator* now sorts the Wire List report that is generated from a Master Designer Schematic or PCB netlist in alpha-numeric order, by net name. As an example, if the nets listed in Wire List report were Net1 through Net100, the order of the nets in the report will be Net1, Net2, Net3, ..., Net10, Net11,....., Net20, Net21, ..., Net 30, Net31,...., Net100.

The general format of the Wire List Report continues to be consistent with previous revisions of Master Designer.

Reference Designation Use Report

Master Designer's Report Generator has been enhanced to include a Reference Designator Use report, which will generate a report from a Schematic netlist, identifying the last used reference designator, not used reference designators, and spare gates.

You can select the Reference Designator Use report from the Schematic Tools Report Generator button. See the figure below for an example of the dialog where the selection of the report is available.

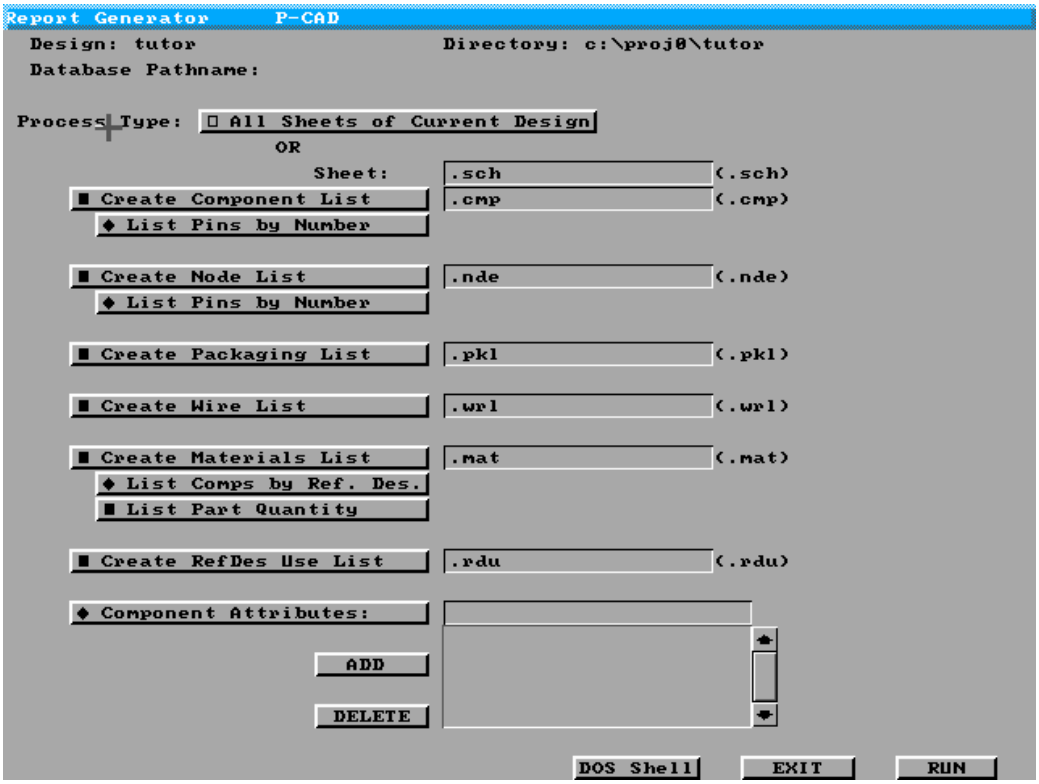


Figure 2.2

When the Create Ref Des Use List button is checked, Report Generator will create a P-CAD Reference Designator Use List report. For an example of this report see figure 2.3 below:

```
%*********************************************************************
%                                                                    *
%     Program  :   pc-form VERSION 8.7                               *
%     Date     :   Jun 24 1997                                       *
%     Time     :   04:42:55 PM                                       *
%     File In  :   t.nlt                                             *
%     File Out :   t.rdu                                             *
%     Format   :   P-CAD Reference Designator Use LIST               *
%                                                                    *
%*********************************************************************


Prefix   Last Used RefDes   Unused RefDes      Spare Gates
------   ----------------   ----------------   ----------------------

C        C3                 C2                 C1/A C1/C C3/B

R        R2

U        U30                U3 U7 U9 U15 U16   U2/C U2/D U10/A U11/B
                            U17 U20 U21 U27    U11/C U11/D
                            U28 U29
```

Figure 2.3

This Reference Designator Use List report describes the following information:

- **Prefix** - This column identifies all of the reference designator prefixes that are contained in the netlist.
- **Last Used Ref Des** - This column identifies the highest numbered reference designator assigned in the netlist for each reference designator prefix.
- **Unused Ref Des** - This column identifies any reference designators that have not been assigned in the netlist.
- **Spare Gates** - This column identifies gates that are not being used in the netlist. A gate is considered to be spare when it is not included in the netlist (placed in the schematic). As an example, if U1 is a 4 gate device and gates U1/A, U1/C, and U1/D are placed in a schematic, gate U1/B would be listed as a spare gate.

The Reference Designator Use report can be generated from *Prevue* using *Design Manager Tool→ Report Generator*, or *Schematic Tool→Report Generator*. It can also be generated from the command line using the -R option. The default filename extension will be *.RDU*.

Autorouter

Activation of the Rip-up Pass

MD 8.7 Autorouter will be enhanced to provide for the activation of the rip-up passes when using the parallel port security device.

System Limits A

This appendix lists the supported system limits for P-CAD Master Designer version 8.6.

Table A-1 lists name length limits. Table A-2 lists maximums. Table A-3 lists other ranges.

Table A-1. Name Length Limits

Parameter	Limit
Attribute keyword	23
Attribute value	255
Bus name (input buffer length)	79
Bus net name length	23
Component name (internal, user assigned)	23
Critical path name length	8
Device name for library member	8
External filename length (including path)	63
Footprint name length	23
Group name length	8
Layer name length	6
Net name (internal, user assigned)	23
Packaging ID (nonhomogeneous parts)	15
Pin name (internal, user assigned)	23
Pin number (alphanumeric)	7
Reference designator length	23
Reference designator prefix length	3
Schematic sheet ID length	4
Section name length	3
Symbol or part external filename, DOS	8.3
Symbol or part external filename, UNIX	255

Table A-2. Maximums

Items	Practical	Theoretical
Aliases per component	1882	
Apertures, number of, Gerber Laser Model	999	
Apertures, number of, Gerber Model 32, 33, 41	24	
Apertures, number of, Gerber RS-274-X	999	
Apertures, number of, MDA Fire 2000	999	
Apertures, size, Gerber Laser Model, RS-274-X, MDA Fire 2000	.500	
Apertures, size, Gerber Model 32, 33, 41	.240	
Association distance (placement)	1000	
Characters in an aperture macro name	12	
Characters in a text string (line)	255	
Clearance for placement	1000	
Components	32,000	(2,147,483,447)
Components per library archive	800	
Filled circle radius	255	
Gate types per package	9	
Gates per package	5000	
Grid spacing	2000	
Input buffer, aperture list, *Environment Set Inner Plane Aperture*	255	
Lattice spacing (placement)	5000	
Lattices for placement	10	
Layer support	100	
LEQ codes	127	
Message character buffer, *Intr*	44	
Nets	64,000	(2,147,483,447)
Picture blocks	250,000	(2,147,483,447)
Pin count, by wire, *Display Control Ratsnest*	512	
Pin types (parts)	1000	
Pins	250,000	(2,147,483,447)
Pins per component	5000	
Polygon aperture width	250	
Polygon clearance value	500	
Polygon + merged void vertices	10,000	
Polygon size (sq. DBUs)	10,240,000	
Polygon vertices	10,000	
Polygonal void vertices	10,000	
Programmable function keys	40	
Repeat copy count	255	
Snap tolerance value	1000	
Stitchable layers	32	
Text size (height)	5000	
Unique components	32,000	(2,147,483,447)
Via types	50	
Wire/line width	255	

Table A-3. Other Ranges

Type	*Range*	*Default*
Angle range, component rotation	-359 to 359	
ASCII key code range for hot keys	33-126	
Bus bits number range	128	
Cartesian coordinates range	-30,000 to 30,000	
Display range, flash size, PCB Editor	4 to 150 DBUs	(60)
Display range, pin size, PCB Editor	4 to 100 DBUs	(40)
Display range, pin size, Schematic Editor	1 to 50 DBUs	(10)
Display range, solder dot, Schematic Editor	1 to 50 DBUs	(10)
Display range, via size, PCB Editor	4 to 100 DBUs	(40)
Pin type range (symbols)	0 to 6, 16 to 63	
Range, histogram routing grid	.01 to 2,000	(50)
Range, histogram signal layers	1 to 20	(2)
Range of D-codes	10-1008	
Range of polygon aperture vertices (or sides)	3-10	
Range of ties in a thermal shape description	1-10	
Scaling factor range, components	1 to 10,000	
Scaling factor range, text	1 to 10,000	

Filename Extensions B

Table B-1. Filename Extensions

Extension	*Description*	*Generated By*
.alt	Netlist input	User (for netlist conversion)
.am	Aperture description macro	User (for PCB aperture table editor)
.apl	Apertures used in design	*PCB Editor*
.apr	Aperture table	*Hardcopy*
.atr	Attribute	*Engineering Change Order*
.asr	Apertures used in plot file	*Hardcopy*
.atr	Attribute	*Engineering Change Order*
.bck	Back-annotation instruction file	*PC-PACK*
.bka	Back-annotation command	*Engineering Change Order*
.bnl	Annotated netlist	*Package Schematic*
.cc	CalComp plotter vector	*Hardcopy*
.cfg	Configuration	Any program run from P-CAD graphic user interface
.cir	Spice file	*Spice Circuit Writer*
.cmd	Packaging command log file	*Package Schematic*
.cmd	Command log file	*Schematic Editor* *PCB Editor* *Symbol Editor* *Part Editor*
.cm$	Backup command log file	*Schematic Editor* *PCB Editor* *Symbol Editor* *Part Editor*
.cmp	Component list report	*Report Generator*
.cth	C. Itoh printer format	*Hardcopy*
.ctl	Autorouter control (strategy)	*Autorouter*
.dbg	Editor debugging	*Schematic Editor* *PCB Editor* *Symbol Editor* *Part Editor*
.dmp	Houston Instruments plotter	*Hardcopy* vector
.drc	Design rule check report	*Design Rules Check*
.drl	Numerically controlled drill data	*Drill*
.dxf	DXF format input/output file	*DXF Reader* *DXF Writer*
.eco	Compare/Analyze phase report	*Engineering Change Order*

Table B-1. Filename Extensions (cont'd)

Extension	*Description*	*Generated By*
.edf	EDIF fomat input/output file	*EDIF Reader* *EDIF Writer*
.eps	Epson format	*Hardcopy*
.erc	Electrical rule check report	*Electrical Rules Check*
.fil	Cross-reference (ASCII)	User (for Package Schematic)
.gbr	Gerber format	*Hardcopy*
.his	Histogram report	*PCB Editor*
.hp	Hewlett-Packard plotter	*Hardcopy*
.hpp	Hewlett-Packard LaserJet printer format	*Hardcopy*
.ibm	IBM printer/plotter format	*Hardcopy*
.icc	Interleaf (CC960) format	*Hardcopy*
.ins	Component insertion	*Auto-Insertion*
.key	Saved function key file	User (created in any graphics editor)
.lgr	Gerber laser photoplotter format	*Hardcopy*
.lgx	RS-274-X photoplotter format	*Hardcopy*
.log	Message log file (DOS)	Any program run from P-CAD graphic user interface and command line
.LOG	Message log file (UNIX)	Any program run from P-CAD graphic user interface and command line
.mac	macro file	User (created in graphics editor or text editor)
.map	DXF translation map file	*DXF Reader* *DXF Writer*
.mat	Materials list report	*Report Generator*
.mda	MDA Fire 2000 photoplotter format	*Hardcopy*
.mfg	NC drill table	*Drill*
.nde	Netlist report	*Report Generator*
.nlc	Report	*Netlist Comparison*
.nlt	Schematic netlist	*Package Schematic*
.nz	Bruning (Nicolet) Zeta plotter	*Hardcopy*
.oki	Okidata printer format	*Hardcopy*
.out	Component name change report	*Engineering Change Order*
.pas	Check pass file	*Design Rules Check (DRC)*

Table B-1. Filename Extensions (cont'd)

Extension	*Description*	*Generated By*
.pbk	Backup PCB	*Engineering Change Order* *Autorouter*
.pdf	Database (ASCII)	*PDIF Writer*
.pin	Pin Information List	*PCB Editor*
.pkg	Packaged PCB database	*Package Schematic*
.pkl	Packaging list report	*Report Generator*
.plc	Placed PCB database	User
.plt	Plot instruction file (binary)	*Schematic Editor* *PCB Editor* *Symbol Editor* *Part Editor*
.plb	PCB part library archive	*Library Maintenance*
.pnl	Packaged PCB netlist (binary)	*Package Schematic*
.ppp	plot device configuration file	*Hardcopy*
.prt	PCB part	*Part Editor*
.ps	Padstack	*Part Editor*
.psc	Postscript format	*Hardcopy*
.pty	Pin type report	*Generate Reports*
.rcf	Autorouter control file	*Autorouter*
.rdu	Ref des use list report	*Report Generator*
.rep	Autorouter report	*Autorouter*
.rhf	Autorouter history file	*Autorouter*
.rpt	Part/symbol pin information table report	*Component Editor*
.rpt	Swap report log file	*PCB Editor*
.rp1	Report	*Convert Netlist to PCB*
.rp2	Report	*Convert Netlist to PCB*
.rp3	Report	*Convert Netlist to PCB*
.rp4	Report	*Convert Netlist to PCB*
.rte	Autorouter extraction	*Autorouter*
.rts	Autorouter solution	*Autorouter*
.rul	Design rules file	*Design Rules Check (DRC)*
.sbk	Backup schematic database	*Schematic Editor*
.sch	Schematic database	*Schematic Editor*
.sdt	Solder dot special symbol	User
.si	Source specification	User
.slb	Schematic symbol library archive	*Library Maintenance*
.ssf	Special symbol file	User (for PCB Editor)
.swr	Improve placement log	*PCB Editor*

Table B-1. Filename Extensions (cont'd)

Extension	*Description*	*Generated By*
.sym	Schematic symbol	*Symbol Editor*
.tbl	NC drill table	*Drill*
.upd	Update command file	*Engineering Change Order (ECO)*
.wrl	Wire list	*Report Generator*
.xnl	Expanded schematic netlist	*Package Schematic*
.xrf	Nodes cross-reference file	*Spice Circuit Writer*

Reserved Words C

This appendix lists DOS reserved device names and attribute keywords.

DOS Reserved Device Names

Don’t use the following DOS reserved device names in filename prefixes:

aux

com1

com2

com3

com4

con

lpt1

lpt2

lpt3

nul

prn

Refer to your DOS manual for more information.

Attribute Keywords

Certain keywords have predefined functions in specific tools. Table C-1 lists the keywords and tools.

Table C-1. Reserved Keywords With Predefined Functions

Keyword	*Tool*
CARD*x* (where *x* is a number)	*SPICE Circuit Writer*
COMPARE=MECH	*Part Editor*, *PCB Editor*
DEVICE	*EDIF Netlist Reader* *EDIF Netlist Writer*
FP	*Part Editor* *PCB Editor*
INCL*x* (where *x* is a number)	*SPICE Circuit Writer*
NET	*Symbol Editor* *Schematic Editor*
PACKAGE	*EDIF Netlist Reader* *EDIF Netlist Writer*
PCERC	*Electrical Rules Check*
PGCONN	*EDIF Netlist Reader* *EDIF Netlist Writer* *Electrical Rules Check*
PINATT*?*	*EDIF Netlist Reader* *EDIF Netlist Writer*
PNAME	*SPICE Circuit Writer*
PREFIX	*Symbol Editor*
PRT	*Symbol Editor* *Schematic Editor* *Package Schematic*
PWGD	*Symbol Editor* *Schematic Editor* *Package Schematic* *Component Editor*
PWGD *i* (where *i* is a number)	*Symbol Editor* *Schematic Editor* *Package Schematic* *Component Editor*
REFDPRE	*SPICE Circuit Writer* *EDIF Netlist Writer*
RVALUE	*Electrical Rules Check*
SHEET	*Symbol Editor* *Schematic Editor* *Electrical Rules Check*
SNAME	*SPICE Circuit Writer*
S$P	*SPICE Circuit Writer*
SPCC	*SPICE Circuit Writer*

Table C-1. Reserved Keywords With Predefined Functions (con't)

Keyword	*Tool*
SPGN	*SPICE Circuit Writer*
SPINIT	*SPICE Circuit Writer*
SPMI	*SPICE Circuit Writer*
SPP*x* (where *x* is a number)	*SPICE Circuit Writer*
SPTI	*SPICE Circuit Writer*
SPVC	*SPICE Circuit Writer*
TARGET	*Auto-Insertion*

Button Menu Trees D

This appendix contains menu trees for button menus that appear in a graphics editor when you choose the *MnBtn* option in the status area.

Symbol Editor

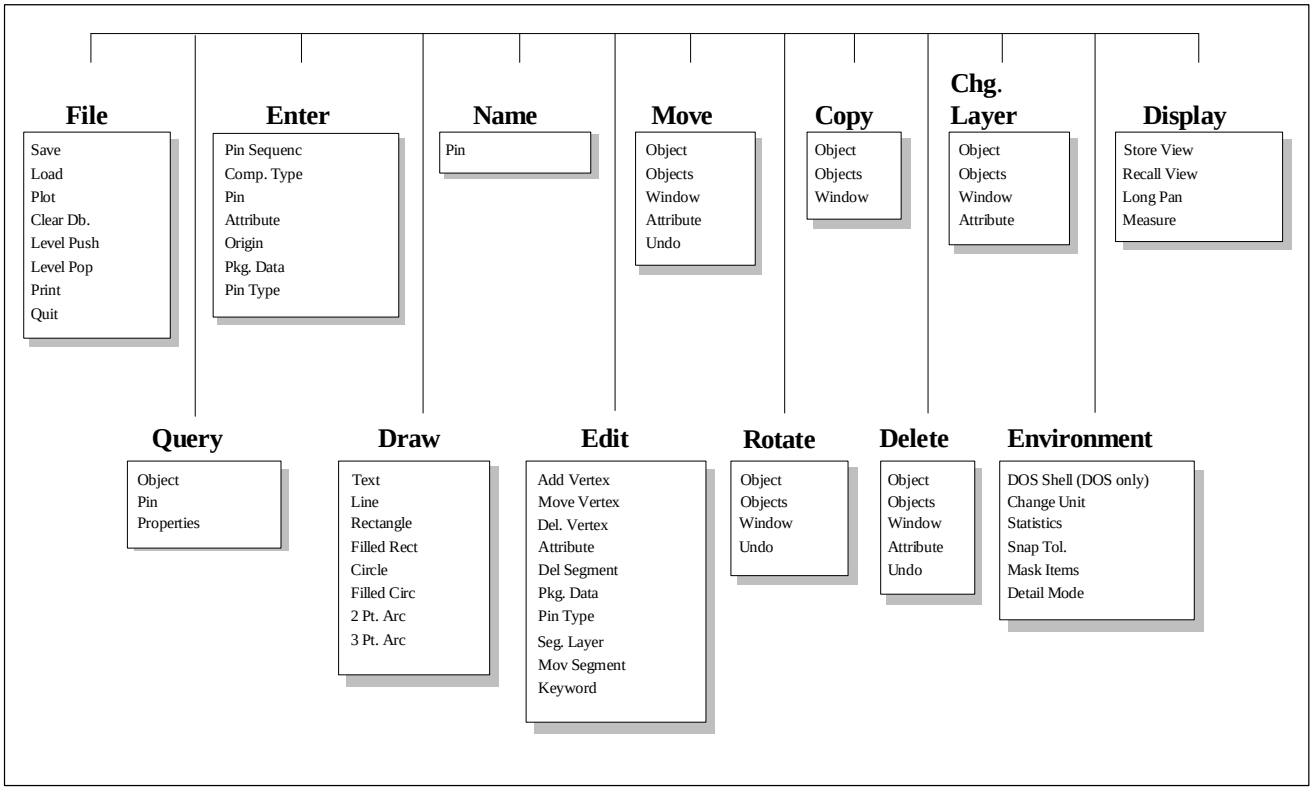


Figure D-1. The Symbol Editor Button Menu Tree

Schematic Editor

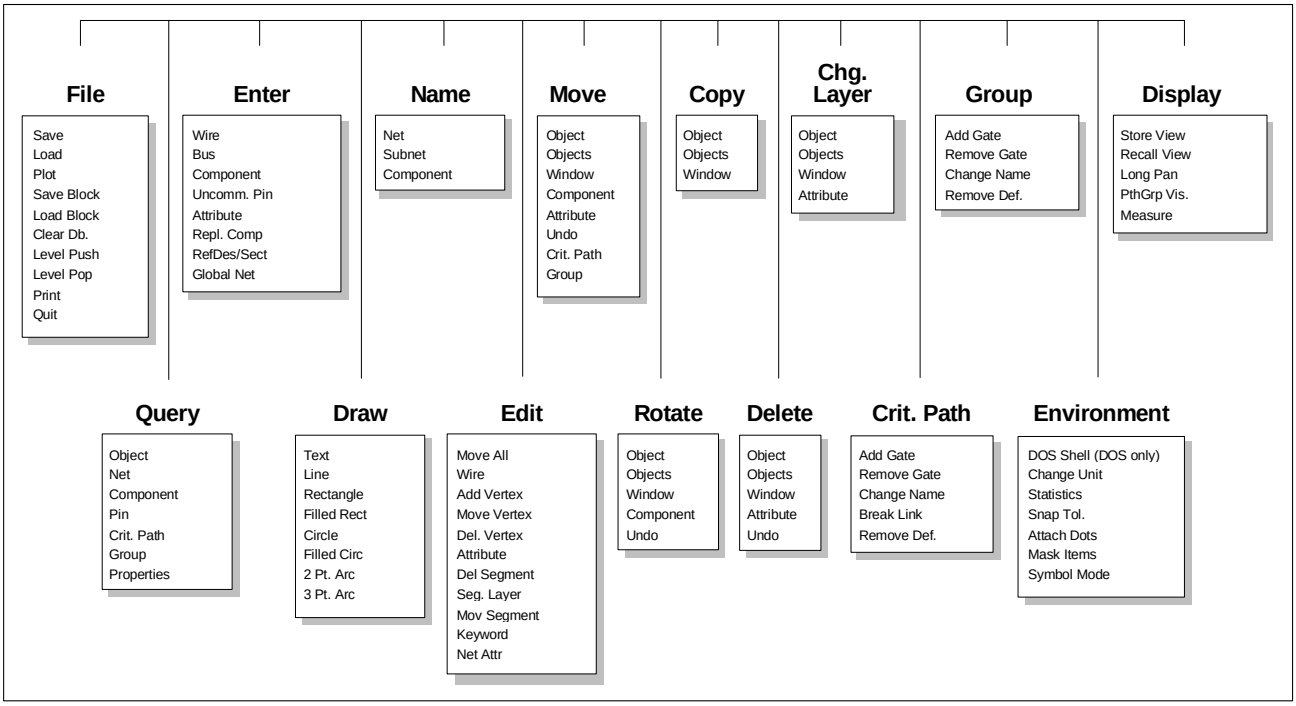


Figure D-2. The Schematic Editor Button Menu Tree

Part Editor

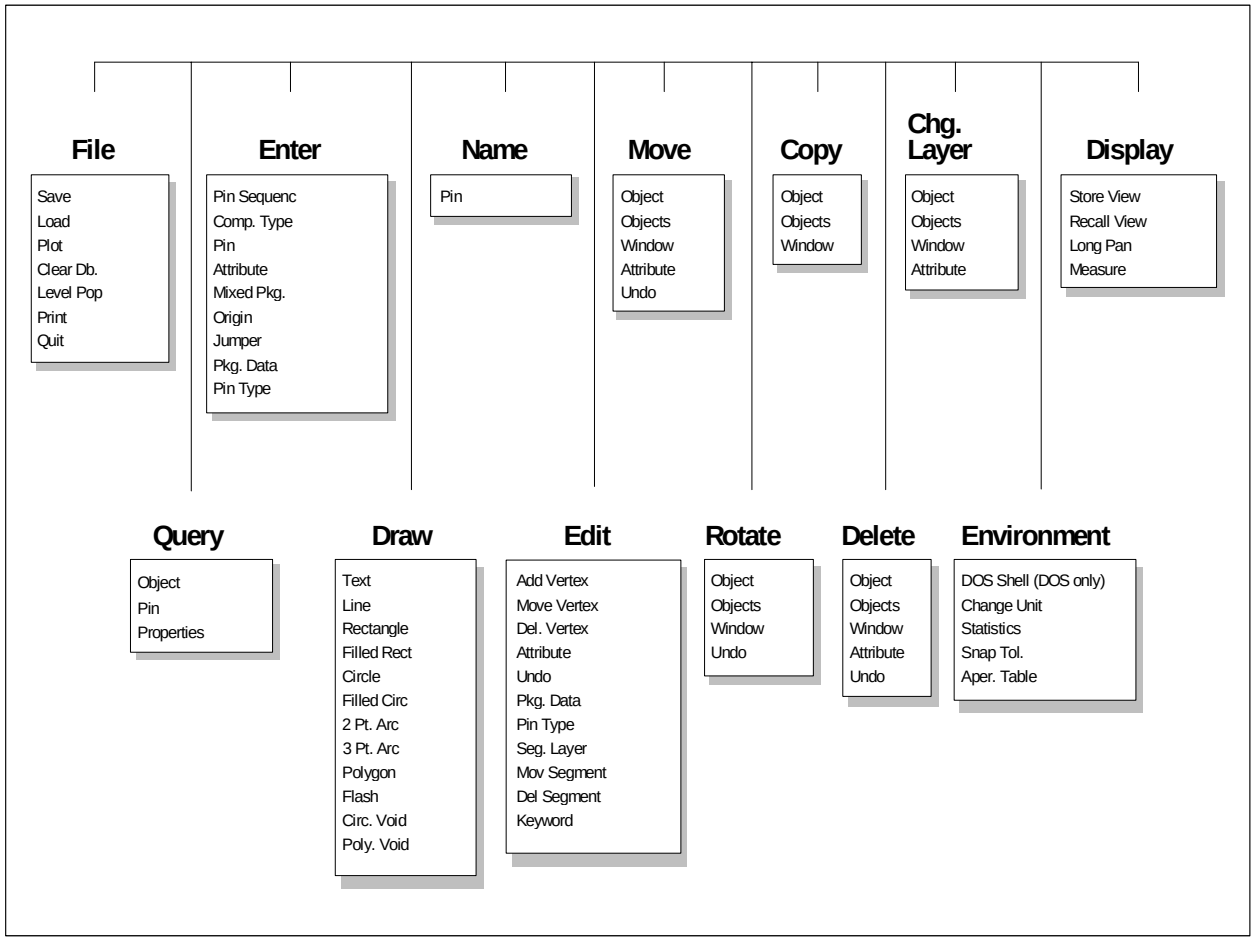


Figure D-3. The Part Editor Button Menu Tree

PCB Editor

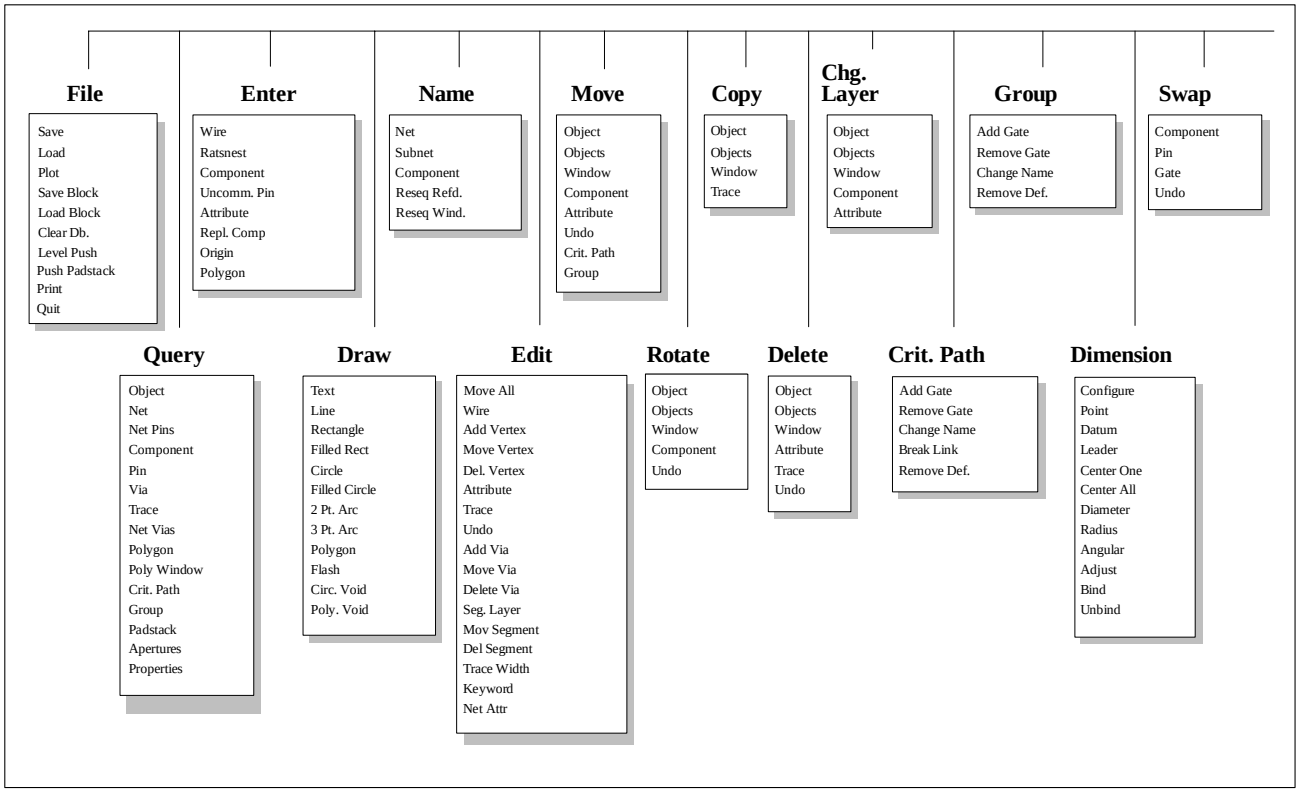


Figure D-4. The PCB Editor Button Menu Tree

PCB Editor (con't)

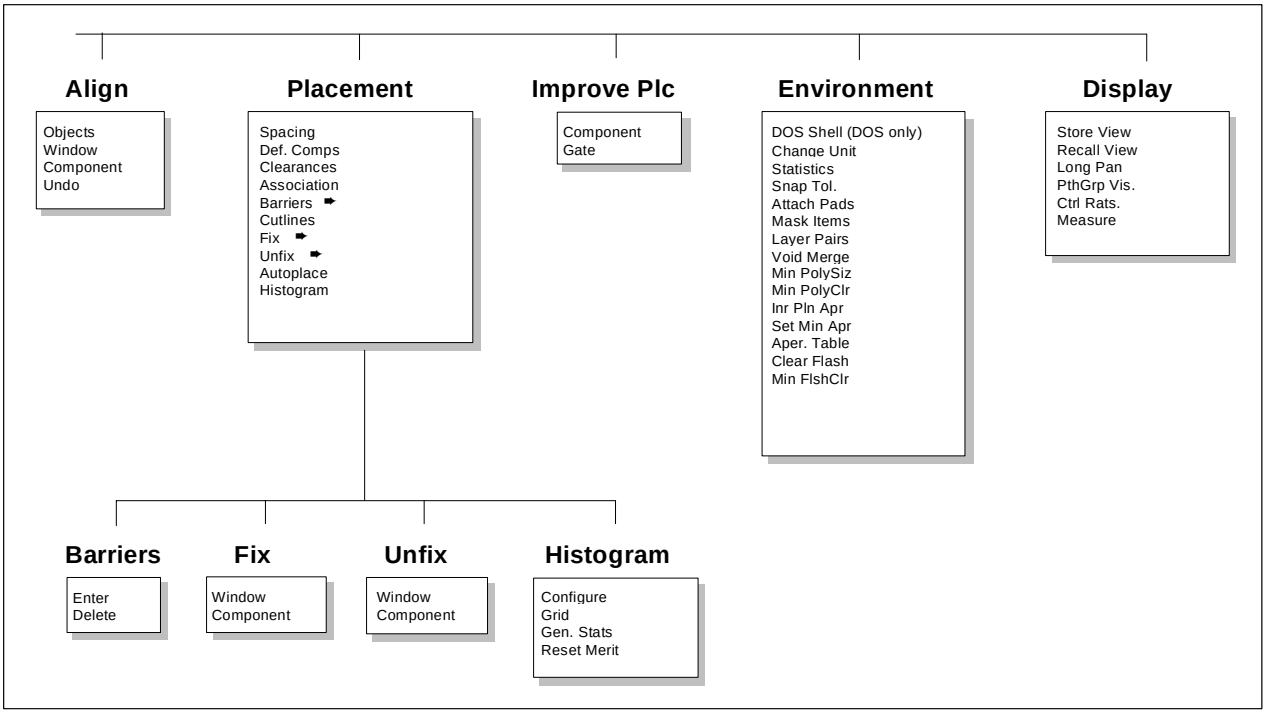


Figure D-4. The PCB Editor Menu Tree (cont'd)

Command Cross Reference E

This appendix contains a command cross-reference, a list of default hot-key functions, and a chart of the keys available for hot-key assignment.

Table E-1 contains a complete cross-reference of MD 5.0 and MD 8.7 graphic editor commands and keyboard-equivalent commands. Refer to the *Command Reference* manual for a complete description of each command.

Commands are listed in alphabetical order by the MD 8.7 commands. Commands that appear in lower case, such as /rctl, have no equivalent menu command for the P-CAD version they're listed under. Keyboard and keyboard-equivalent commands are not case-sensitive.

Table E-1. Command Cross Reference

MD 5.0 Command	***MD 8.7 Command***	***MD 8.7 Keyboard Equivalent***
a (wire bend hot key)	*a*	*a*
a	*a*	*/bend*
n/a	/ably	/ably
ALGN	*Align→Components*	*ALGN*
ALGN/IDEN	*Align→Objects*	*ALGN/IDEN*
ALGN/UNDO	*Align→Undo*	*ALGN/UNDO*
ALGN/WIN	*Align→Window*	*ALGN/WIN*
/alyr	/alyr	*/alyr*
ATTR/DATR	*Delete→Attribute*	*DEL/ATRB*
c (curved wire hot key)	*c*	*c*
c	*c*	*/wcrv*
/cfil	*/cfil*	*/cfil*
n/a	*Chg. Layer→Attribute*	*CLYR/ATRB*
CLYR/COMP	*Chg. Layer→Component*	*CLYR/COMP*
FLIP	*Chg. Layer→Component*	*CLYR/COMP*
CLYR	*Chg. Layer→Object*	*CLYR*
CLYR/IDEN	*Chg. Layer→Objects*	*CLYR/IDEN*
CLYR/WIN	*Chg. Layer→Window*	*CLYR/WIN*
COPY	*Copy→Object*	*COPY*
COPY/IDEN	*Copy→Objects*	*COPY/IDEN*
COPY/TRCE	*Copy→Trace*	*COPY/TRCE*
COPY/WIN	*Copy→Window*	*COPY/WIN*
/cpos	*/cpos*	*/cpos*
CPTH/TAG	*Crit. Path→Add Gate*	*CPTH/TAG*
CPTH/UNLK	*Crit. Path→Break Link*	*CPTH/UNLK*
CPTH/RNAM	*Crit. Path→Change Name*	*CPTH/RNAM*
CPTH/RSET	*Crit. Path→Remove Definition*	*CPTH/RSET*
CPTH/UTAG	*Crit. Path→Remove Gate*	*CPTH/UTAG*
ATTR/DATR	*Delete→Attribute*	*DEL/ATRB*
DEL	*Delete→Object*	*DEL*
DEL/IDEN	*Delete→Objects*	*DEL/IDEN*
DEL/TRCE	*Delete→Trace*	*DEL/TRCE*
DEL/UNDO	*Delete→Undo*	*DEL/UNDO*
DEL/WIN	*Delete→Window*	*DEL/WIN*

n/a	*Dimension→Adjust*	*ADIM/AJST*

Table E-1. Command Cross Reference (cont'd)

MD 5.0 Command	MD 8.7 Command	MD 8.7 Keyboard Equivalent
n/a	Dimension→Angular	ADIM/ANGL
n/a	Dimension→Bind	ADIM/DGRP
n/a	Dimension→Center All	ADIM/CALL
n/a	Dimension→Center One	ADIM/CONE
n/a	Dimension→Configure	ADIM/CNFG
n/a	Dimension→Datum	ADIM/DATM
n/a	Dimension→Diameter	ADIM/DIAM
n/a	Dimension→Leader	ADIM/LEAD
n/a	Dimension→Point	ADIM/PONT
n/a	Dimension→Radius	ADIM/RADI
n/a	Dimension→Unbind	ADIM/UGRP
/rctl	Display→Control Ratsnest	RCTL
/lpan	Display→Long Pan	LPAN
/rulr	Display→Measure	RULR
/pctl	Display→Path/Group Visiblity	PCTL
RCL	Display→Recall View	RCL
n/a	Display→Recall View 1	RC1
n/a	Display→Recall View 2	RC2
n/a	Display→Recall View 3	RC3
n/a	Display→Recall View 4	RC4
n/a	Display→Recall View 5	RC5
n/a	Display→Recall View 6	RC6
n/a	Display→Recall View 7	RC7
n/a	Display→Recall View 8	RC8
n/a	Display→Recall View 9	RC9
n/a	Display→Recall View 10	RC10
STO	Display→Store View	STO
n/a	Display→Store View 1	ST1
n/a	Display→Store View 2	ST2
n/a	Display→Store View 3	ST3
n/a	Display→Store View 4	ST4
n/a	Display→Store View 5	ST5
n/a	Display→Store View 6	ST6
n/a	Display→Store View 7	ST7
n/a	Display→Store View 8	ST8
n/a	Display→Store View 9	ST9
n/a	Display→Store View 10	ST10
DRAW/ARC	Draw→2-Point Arc	DRAW/ARC
DRAW/ARCP	Draw→3-Point Arc	DRAW/ARCP
DRAW/CIRC	Draw→Circle	DRAW/CIRC
DRAW/CVOD	Draw→Circular Void	DRAW/CVOD
n/a	Draw→Filled Circle	DRAW/FCIR
DRAW/FREC	Draw→Filled Rectangle	DRAW/FREC
DRAW/FLSH	Draw→Flash	DRAW/FLSH
DRAW/LINE	Draw→Line	DRAW/LINE
DRAW/POLY	Draw→Polygon	DRAW/POLY
DRAW/PVOD	Draw→Polygonal Void	DRAW/PVOD
DRAW/RECT	Draw→Rectangle	DRAW/RECT
DRAW/TEXT	Draw→Text	DRAW/TEXT
n/a	/dly	/dly
/drc	/drc	/drc
/dyn	/dyn	/dyn
EDIT/ADDV	Edit→Add Vertex	EDIT/ADDV
EDIT/AVIA	Edit→Add Via	EDIT/AVIA

ATTR/SCHG	*Edit→Attribute*	*EDIT/ATRB*
EDIT/DELS	*Edit→Delete Segment*	*EDIT/DELS*

Table E-1. Command Cross Reference (cont'd)

MD 5.0 Command	MD 8.7 Command	MD 8.7 Keyboard Equivalent
n/a	Edit→Delete Trace	EDIT/TRCE
EDIT/DELV	Edit→Delete Vertex	EDIT/DELV
EDIT/DVIA	Edit→Delete Via	EDIT/DVIA
n/a	Edit→Keyword	EDIT/KWD
EDIT/MOVA	Edit→Move All	EDIT/MOVA
EDIT/MOVS	Edit→Move Segment	EDIT/MOVS
EDIT/MOVV	Edit→Move Vertex	EDIT/MOVV
EDIT/MVIA	Edit→Move Via	EDIT/MVIA
n/a	Edit→Net Attr	EDIT/NATR
SCMD/EPKG	Edit→Packaging Data	EPKG
SCMD/EPNL	Edit→Packaging Data	EPNL
n/a	Edit→Pin Type	EDIT/SPAT
EDIT/LAYS	Edit→Segment Layer	EDIT/LAYS
n/a	Edit→Trace Width	EDIT/TWID
/undo	Edit→Undo Delete Segment	UNDO
EDIT/WIRE	Edit→Wire	EDIT/WIRE
ATTR/ACOM	Enter→Attribute	ENTR/ATRB
ENTR/ORG	Enter→Board Origin	ENTR/ORG
ENTR/BUSB	Enter→Bus	ENTR/BUSB
ENTR/COMP	Enter→Component	ENTR/COMP
SCMD/SCAT	Enter→Component Type	ENTR/SCAT
SCMD/SNAT	Enter→Global Net	ENTR/SNAT
SCMD/JMPR	Enter→Jumper	ENTR/JMPR
SCMD/NPKG	Enter→Nonhomogeneous Pkg	ENTR/NPKG
ENTR/ORG	Enter→Origin	ENTR/ORG
SCMD/PNLC	Enter→Packaging Data	ENTR/PNLC
SCMD/SPKG	Enter→Packaging Data	ENTR/SPKG
ENTR/PIN	Enter→Pin	ENTR/PIN
ENTR/SEQ	Enter→Pin Sequence	ENTR/SEQ
SCMD/SPAT	Enter→Pin Type	ENTR/SPAT
ENTR/POLY	Enter→Polygon	ENTR/POLY
ENTR/RATN	Enter→Ratsnest	ENTR/RATN
NAME/REFD	Enter→Ref.Des.& Section	ENTR/PNUM
SCMD/PNUM	Enter→Ref.Des.& Section	ENTR/PNUM
/repl	Enter→Replace Component	REPL
ATTR/ACOM	Enter→Sheet Number	ENTR/SHID
ENTR/UCOM	Enter→Uncommit Pin	ENTR/UCOM
ENTR/WIRE	Enter→Wire	ENTR/WIRE
SCMD/LPAR	Environment→Assign Layer Pairs	LPAR
SCMD/GSSF	Environ.→Attach Cust.Sold.Dots	GSSF
SCMD/GSSF	Environment→Attach Padstacks	GSSF
SCMD/UNIT	Environment→Change Units	UNIT
SYS/DOS	Environment→DOS Shell	DOS
DETL	Environment→Detail Mode	DMOD
SYS/STAT	Environment→Display Statistics	STAT
n/a	Environment→Edit Aperture Table	ATE
SCMD/SIPC	Environ.→Inner Plane Apertures	SIPC
/mask	Environment→Mask Items	MASK
/msk	Environment→Mask Items	MSK
SCMD/VMRG	Environ.→Merge Polygon Voids	VMRG
n/a	Environ.→Merge Voids by Layer	VMBL
n/a	Environ.→Merge Voids by Poly	VMBP
SCMD/PSIZ	Environment→Min. Polygon Size	PSIZ

SCMD/PCLR	*Envir.→Polygon Wire Clearance*	*PCLR*
n/a	*Environ.→Set Minimum Aperture*	*MAPW*
/sgat	*Environment→Set Snap Tolerance*	*SGAT*

Table E-1. Command Cross Reference (cont'd)

MD 5.0 Command	MD 8.7 Command	MD 8.7 Keyboard Equivalent
SYMB	Environment→Symbol Mode	SMOD
/exe	/exe	/exe
/exec	/exec	/exec
n/a	/fast	/fast
FILE/ZAP	File→Clear Database	FILE/ZAP
SYS/PLOT	File-Create Plot File	FILE/PLOT
FILE/POP	File→Level Pop	FILE/POP
LEVL/POP	File→Level Pop	FILE/POP
FILE/PUSH	File→Level Push	FILE/PUSH
LEVL/PUSH	File→Level Push	FILE/PUSH
FILE/LOAD	File→Load	FILE/LOAD
FILE/BKLD	File→Load Block	FILE/BKLD
n/a	File→Print	FILE/PRINT
n/a	File→Push Padstack	FILE/PUPS
SYS/QUIT	File→Quit	FILE/QUIT
FILE/SAVE	File→Save	FILE/SAVE
FILE/BKSV	File→Save Block	FILE/BKSV
/fit	Fit View	FIT
/fitv	Fit View	FITV
n/a	/grid	/grid
GRP/TAG	Group→Add Gate	GRP/TAG
GRP/RNAM	Group→Change Name	GRP/RNAM
GRP/RSET	Group→Remove Definition	GRP/RSET
GRP/UTAG	Group→Revome Gate	GRP/UTAG
IMPR/COMP	Improve Plc→Components	IMPR/COMP
IMPR/GATE	Improve Plc→Gates	IMPR/GATE
/intr	/intr	/intr
/lang	/lang	/lang
/lsty	/lsty	/lsty
/lwid	/lwid	/lwid
/lyrn	/lyrn	/lyrn
n/a	Last View	PREV
/mac	/mac	/mac
n/a	Main Menu	/main
/mend	/mend	/mend
n/a	/menu	/menu
MOVE/ATRB	Move→Attribute	MOVE/ATRB
MOVE/COMP	Move→Component	MOVE/COMP
MOVE/APTH	Move→Critical Path	MOVE/CPTH
MOVE/AGRP	Move→Group	MOVEAGRP
MOVE	Move→Object	MOVE
MOVE/IDEN	Move→Objects	MOVE/IDEN
MOVE/UNDO	Move→Undo	MOVE/UNDO
MOVE/WIN	Move→Window	MOVE/WIN
NAME/COMP	Name→Component	NAME/COMP
NAME/NET	Name→Net	NAME/NET
NAME/PIN	Name→Pin	NAME/PIN
NAME/RSEQ	Name→Reseq. Ref. Desd	NAME/RSEQ
n/a	Name→Reseq. Window	NAME/SEQW
NAME/SUBN	Name→Subnet	NAME/SUBN
/ofly	/ofly	/ofly
n/a	/only	/only
n/a	/osel	/osel
PAN	Pan	PAN
Pan Bars	Shift Pan	[SHIFT]+[Btn 1]

/pdel /pend	/pdel /pend	/pdel /pend

Table E-1. Command Cross Reference (cont'd)

MD 5.0 Command	MD 8.7 Command	MD 8.7 Keyboard Equivalent
/pkey	/pkey	/pkey
/pkld	/pkld	/pkld
/pksv	/pksv	/pksv
/pzap	/pzap	/pzap
PLCE	Placement→Automatic Placement	PLCE
BARR/DEL	Placement→Define Barriers→Delete	BARR/DELB
BARR/DELB	Placement→Define Barriers→Delete	BARR/DELB
BARR	Placement→Define Barriers→Enter	BARR
ASSC	Placement→Define Lattices→ Associate Discretes	ASSC
LATC	Placement→Define Lattices→ Define Components	LATC
LATP	Placement→Define Lattices→ Enter Spacing	LATP
CLR	Placement→Define Lattices→ Specify Clearance	CLR
CUT	Placement→Enter Cutlines	CUT
FIX	Placement→Fix→Component	FIX
FIX/WIN	Placement→Fix→Window	FIX/WIN
HIST/CNFG	Placement→Histogram→Configure	HIST/CNFG
HIST/GRID	Placement→Histogram→Define Grid	HIST/GRID
HIST/STAT	Placement→Histogram→ Generate Statistics	HIST/STAT
HIST/RSET	Placement→Histogram→ Reset Merit Factor	HIST/RSET
UNFX	Placement→UnFix→Component	UNFX
UNFX/WIN	Placement→UnFix→Window	UNFX/WIN
n/a	Query→Apertures	QRY/APRS
QRY/COMP	Query→Component	QRY/COMP
QRY/APTH	Query→Critical Path	QRY/APTH
QRY/AGRP	Query→Group	QRY/AGRP
QRY/NET	Query→Net	QRY/NET
QRY/NPIN	Query→Net Pins	QRY/NPIN
n/a	Query→Net Vias	QRY/NVIA
QRY	Query→Object	QRY
QRY/PSTK	Query→Padstack	QRY/PSTK
QRY/PIN	Query→Pin	QRY/PIN
n/a	Query→Polygon	QRY/POLY
n/a	Query→Polygon Window	QRY/PWIN
n/a	Query→Properties	QRY/PROP
n/a	Query→Trace	QRY/TRCE
QRY/VIA	Query→Via	QRY/VIA
/rcmn	n/a	n/a
RCL	Recall View	RCL
REDR	Redraw	REDR
n/a	/regn	/regn
/resu	/resu	/resu
n/a	/rnsp	/rsnp
ROT/COMP	Rotate→Component	ROT/COMP
ROT	Rotate→Object	ROT
ROT/IDEN	Rotate→Objects	ROT/IDEN
ROT/UNDO	Rotate→Undo	ROT/UNDO
ROT/WIN	Rotate→Window	ROT/WIN
/stgl	/stgl	/stgl
SWAP/COMP	Swap→Component	SWAP/COMP

Table E-1. Command Cross Reference (cont'd)

MD 5.0 Command	***MD 8.7 Command***	***MD 8.7 Keyboard Equivalent***
SWAP/GATE	*Swap→Gate*	*SWAP/GATE*
SWAP/PIN	*Swap→Pin*	*SWAP/PIN*
SWAP/UNDO	*Swap→Undo*	*SWAP/UNDO*
n/a	*/tmod*	*/tmod*
/ucp	*/ucp*	*/ucp*
/undo	*/undo*	*/undo*
VLYR	*View Layer*	*VLYR*
VSAV	*n/a*	n/a
VWIN	*View Window*	*VWIN*
/wait	*/wait*	*/wait*
x (full screen cursor hot key)	*x* (full screen cursor hot key)	*x*
x	*x*	*/xhar*
ZIN	*Zoom In*	*ZIN*
ZOUT	*Zoom Out*	*ZOUT*

Table E-2 describes the default hot-key assignments for MD 8.7 commands or functions. Hot keys for subcommands will be recognized only if the main menu command has already been executed. For example, to move a wire vertex, press **E** to activate *Edit*, then **G** to execute the *Move→Vertex* command.

Table E-2. Default Hot Key Key Functions

Key	*Command or Function*
a	Toggle wire bend
b	*Delete Vertex*
c	Toggle curved wire (PCB Editor only)
d	*Delete*
e	*Edit*
f	*File*
g	*Move Vertex*
h	*Delete Segment*
i	*Wire*
j	*Move Segment*
k	*Component*
l	*Load*
m	*Move*
n	*Enter*
o	*Objects*
p	*Pan*
q	*Query*
r	*Rotate*
Alt+R	rotate clockwise
Ctrl+R	rotate counterclockwise
Shft+s	Snap to Object (Measure)
s	*Save*
t	*Trace (PCB Editor only)*
u	*Undo*
v	*Add Vertex*
w	*Window*
x	Cursor type (regular, vertical or diagonal cross hair)
y	*Segment Layer*
z, +,=	*Zoom In*
Z*, -	*Zoom Out*

* Note: Except for Z only lower-case keys have default functions.

Table E-3 contains a chart of the ASCII characters available for hot-key assignment in MD 8.7.

Table E-3. ASCII Key Codes Available For Hot Key Assignment

		ASCII Key Codes									
↓ Left Digit	Right Digit	**0**	**1**	**2**	**3**	**4**	**5**	**6**	**7**	**8**	**9**
3					!	”	#	$	%	&	’
4		(	)	*	+	,	-	.	n/a	0	1
5		2	3	4	5	6	7	8	9	:	;
6		<	=	>	?	@	A	B	C	D	E
7		F	G	H	I	J	K	L	M	N	O
8		P	Q	R	S	T	U	V	W	X	Y
9		Z	[	\	]	^	-	‘	a	b	c
10		d	e	f	g	h	I	j	k	l	m
11		n	o	p	q	r	s	t	u	v	w
12		x	y	z	{	|	}	~			

ASCII key codes range from 00 to 127. The following characters or ASCII key codes aren’t available for hot-key assignment:

- 00 to 31 and 127 aren’t available because they’re nonprintable control characters
- 47 (/) is reserved for initiating keyboard commands
- 32 represents a blank and isn’t available for user assignment
- **Shift**, **Ctrl**, **Alt**, ◊, **<**, and **>** can’t be used to assign commands or functions to P-CAD hot keys

This appendix lists a command cross-reference, default hot key functions, and a chart of keys available for hot keys.

Index